SIMULATION OF CENTER LOCATION OF FLUID FLOW IN SHEAR DRIVEN CAVITY USING LATTICE BOLTZMANN METHOD

WAN MOHD HANIF BIN OTHMAN

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

ABSTRACT

The study of center location of lid driven cavity using lattice Boltzmann method is all about the of the fluid dynamic base on the simulation and prediction the flow. The study is base on steady flow and transient flow using Lattice Boltzmann method to understant the capability of Lattice Boltzmann. The simulation of Lattice Boltzmann is using FOTRAN software. The result is compare with Ghia et al to validate the stream function is in good arrangement and also can determine the capability of LBM. The result have been compare with simulation lid driven cavity using ANSYS(FLUENT) using navier stoke solver at the transient flow in Reynolds number 100 to 10000. From the simulation, the lattice Boltzmann method is capable at Reynolds number below than 7500.

ABSTRAK

Kajian simulasi tentang pusaran utama didalam rongga segiempat sama yang tudungnya digerakkan utk mengkaji pergerakan bendalir menggunakan kaedah kekisi Boltzmann. Dalam kajian bendalir secara simulasi, ramalan pergerakan berdasarkan kaedah berangka digunakan. Simulasi ini adalah untuk mengkaji kebolehupayaan kaedah kekisi Boltzmann berbanding kaedah lain. hasil dari simulasi ini telah dibandingkan menggunakan hasil simulasi Ghia et al untuk perbandingan struktur aliran fungsi berada dalam keadaan baik. Simulasi juga dijalankan menggunakan perisian ANSYS (FLUENT) iaitu berasaskan navier stoke dalam keadaan bergerak untuk mengkaji pergerakan pusat utama. Daripada simulasi ini, kekisi lattice Boltzmann berupaya menunjukkan hasil baik dari simulasi semasa bergerak, atau simulasi semasa akhir pergerakan pada nombor Reynolds sebelum 7500.

TABLE OF CONTENTS

CHAPTER 3 METHODOLOGY

CHAPTER 4 RESULTS AND DISCUSSION

CHAPTER 5 CONCLUSION AND RECOMENDATION

LIST OF TABLE

LIST OF FIGURES

LIST OF SYMBOLS

u	Velocity vector
t	Time
$F(x,c,t)$	Density distribution function
f_i	Discretised Density Distribution Function
f_i^{eq}	Discretised Equilibrium Density Distribution Function
x	Space vector
C	Micro Velocity Vector
a	Acceleration
T	Temperature
k_b	Boltzmann Constant
g	Particle's relatice velocity
	Gravity Force
R	Gas Constant
D	Dimension
W_k	Weight coefficient
U	Top wall horizontal velocity
u	Horizontal velocity
v	Vertical velocity
n	Mole number
V	Volume
υ	Kinematic shear viscosity
ρ	Density
Ω	Collision operator
ψ	Stream function
τ	Average time between excessive collision
Re	Reynolds number
Nu	Nusselt number
Pr	Prandtl Number
Ra	Rayleigh number

LIST OF ABBREVIATIONS

BGK	Bhatnagar – Gross - Krook
CFD	Computional fluid dynamic
D2Q9	Two dimentions nine velocity
FD	Finite Difference
LB	Lattice Boltzmann
NSE	Navier stoke equation
FEM	Finite Element Method
LBM	Lattice Boltzmann method
LBE	Lattice Boltzmann Equation
LGA	Lattice gas Approsh

CHAPTER 1

INTRODUCTION

1.1 LID DRIVEN CAVITY FLOW (LDCF)

Understanding of fluid dynamic is very important in most branch of engineering especially in mechanical engineering. The fluid dynamic touches in many aspect of daily life such as air conditioning system to comfort people in the room and the simple thing such make a tea in a cup. To understand the fluid dynamic, we need to visualize the movement or flow of the fluid. The fluid mechanics need to visualize with time and space for more understanding and we can realize with the problem on it. The visual of fluid mechanic can be produce from the experiment that is high cost and the simulation by software which is almost accurate and low cost.

The shear driven cavity or also called lid-driven cavity flow is not only technically important to solve fluid flow problem but also the great scientific interest because it displays almost all fluid mechanical phenomena in the simplest of geometrical settings (Ghia et al 1982; D.A. Parumal, A.K Dass, 2009). Lid-driven cavity flow problem also has received considerable attention mainly because of its geometric simplicity, physical abundance, and its close relevance to some fundamental engineering (T. P. Chiang, W. H. Sheu 1997). The simplicity of the geometry of the cavity flow makes the problem easy to code and apply boundary conditions. Even though the problem looks simple in many ways, the flow in a cavity retains all the flow physics with counter rotating vortices appear at the corners of the cavity. (E. Erturk 2009).

The study about 2-D driven cavity flow problem is discussed in details in terms of physical and mathematical and also numerical aspects (E. Erturk 2009). The lid-driven-cavity problem is one of the most important benchmarks for numerical Navier–Stokes solvers. It can be subject grouped into three categories, in the first category of studies, steady solution of the driven cavity is sought. In these types of studies the numerical solution of steady incompressible Navier-Stokes equations are presented at various Reynolds numbers such as the results from Ghia et al 1982 and Erturk et al 2005 . In the second category of studies, the bifurcation which is the place where something divides into two branches, of the flow in a driven cavity from a steady regime to an unsteady regime is studied. In these studies a hydrodynamic stability analysis is done and the Reynolds numbers at which a Hopf bifurcation occurs in the flow are presented. The results from second category are from Fortin et al., Gervais et al., Sahin and Owens. In the third category of studies, the transition from steady to unsteady flow is studied through a Direct Numerical Simulation (DNS) and the transition Reynolds number is presented. The paper in the third category is such as Che Sidek, N.A and Nik Mu'tasim, M.A (2009). There are also the study in three dimensions (3D) of lid driven cavity that is from S. Albensoeder and H.C. Kuhlmann.

Ghia et al.1982 were among the first to publish benchmark data on the lid driven cavity flow. These classical papers are frequently referenced even today (S. Albensoeder, H.C. Kuhlmann). This paper was using Navier- stokes equation and a multigrid method in 2-D simulation. The Reynold number that been used for this paper were from 100 to10000 with meshes consisting of as many as 257 × 257 grid points. This paper was group into first category according to E. Erturk 2009.

A lid-driven cavity consists in a cavity bounded by solid walls. One of these walls is allowed to translate along itself, dragging the fluid which adheres to it (S. Nguyen et al 2006). When the lid was moving in u velocity in figure 1, it will cause on the flow of fluid. Basically, the governing equations for 2D deep cavity flow are developed from Navier-Stokes equation and continuity equation. (Mat Sahat, M.I , Che Sidek, N.A , 2010)

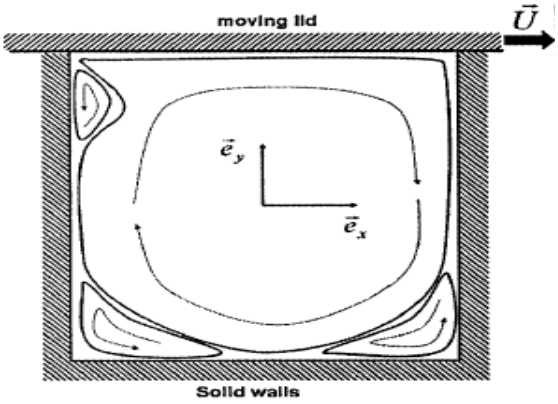

Figure1.0: Lid-driven cavity configuration

Source: S. Nguyen et al 2006

1.2 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) that was developed over 40 years ago by engineers and mathematicians. The development of CFD because they want to solve heat and mass transfer problems in aeronautics, vehicle aerodynamics, chemical engineering, nuclear design and safety, ventilation and industrial design. The development of this technology in the 1950s and 1960s made such research possible, and CFD was one of the first areas to take advantage of the newly emergent field of scientific computing. In the process, it was soon realized that CFD could be an alternative to physical modeling in many areas of fluid dynamics, with its advantages of lower cost and greater flexibility.

Computational fluid dynamics is therefore an area of science made possible by and fundamentally linked to, computing. Its growth has paralleled that of computer power and availability, and as we move into an age of cheap, powerful desktop computing it is now possible, with a little knowledge, to run large and complex 3D simulations on an average personal computer.

However, most research advances in CFD continue to originate in the aeronautics and industrial design communities as a result of the significant investment levels available in these areas. In such cases it may be possible to characterize the complete set of process mechanisms that exist and also obtain good experimental data for model validation. Major research questions, therefore, concern improvements to the quality of the numerical solution, the scales of flow resolved by the model for fixed computational costs and the representation of sub-grid-scale processes such as turbulence.

The applications of CFD are used as tools for research, design, education, Automotive, Sports and many other fields. In this thesis, the focus is based on usefulness of CFD base on comparing Navier stoke Equation and Latice Boltzmann Method.

There are two type of CFD simulation which is the numerical and the other one is ready to use software. The software ready to use such as FLUENT© is very easy to use and infinite type of flow problem with many variables can be easily solved but there are disadvantages such as the user probably doesn't know to the depth about the formulations that has been applied, the assumptions and a lot more. This software normally used for the practical application which the complicated geometry and conditions and used navier stoke as a solver. Despite that, this software is based on the numerical method but it is not being revealed. It purposes is solely to reduce the tough part and to make it user friendly (Mat Sahat, M.I 2010). The CFD simulation which is used numerical method software such as FORTRAN, C++, Matlab need the user create the codes and understand very well the formulation, the assumption, boundary conditions and others. This style of simulation usually applicable for knowledge sharing as many publications spawn everyday with new type of method for example the Lattice Boltzmann method that been used for this paper, Bifurcation method and more, claiming the method is among the best through various comparison and validation with the earlier or the classical method like Ghia et al. 1982. The simulation requires the creator to be well-verse in programming software that been used.

1.3 PROBLEM STATEMENT

The lid driven cavity problem is one of the most important benchmarks for numerical Navier–Stokes solvers. From this statement we can conclude that the result from the Navier-Strokes from lid driven cavity problem can be compare with other analysis base on lid driven cavity flow. This result will compare with lattice Boltzmann method. The critical for this analysis may be from the translation flow because in this regime the flow will change from laminar to turbulent. This study will provide information or subject to use of LBM and its extent toward solving fluid flow analytically.

1.4 OBJECTIVE

The objective for this analysis is to simulate the flow in the lid driven cavity by using lattice Boltzmann method. The analysis will use FORTRAN. The result from lattice Boltzmann method will be compare with Navier Stoke equation that is from Ghia et al 1982 and ANSYS (FLUENT©).

1.5 SCOPE

i. To simulate the flow in the lid driven cavity by using lattice Boltzmann method.
ii. The result from lattice Boltzmann method will be compare with Navier Stoke equation using ANSYS (FLUENT©)
iii. To understand the capability of LBM method in fluid flow based on qualitative study.

1.6 SCOPE OF ANALYSIS

i. In this analysis, the simulation will apply Lattice Boltzmann using FORTRAN language. The simulation is used to analyze the stream function with different Reynolds number that is from 100 to 10000.

ii. The boundary condition is no slip condition. The mashes size of cavity is 301×301.

iii. Validation is done using LBM, LDC in steady state condition.

iv. The result of this analysis will be compare with the result from experiment from Ghia et. al. (1982).

v. The result for LBM will compare to FLUENT software using Navier Stroke equation in transient flow condition in LDC.

CHAPTER 2

LITERATURE REVIEW

2.1 NAVIER STOKES EQUATION

The Navier–Stoke equation is well known equation for fluid dynamic describes the motion of fluid substances. These equations arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term.

The mathematical relationship governing fluid flow is the continuity equation, from general differential equation from conservation of mass,

$$\frac{\delta\rho}{\delta t} + \vec{\nabla}.\left(\rho\vec{\mathrm{V}}\right) = 0 \tag{2.1}$$

The mathematical relationship governing fluid flow is the famous continuity equation

$$\nabla.u = 0 \tag{2.2}$$

And the Navier-strokes equation

$$\frac{\delta u}{\delta t} + u.\nabla u = -P + v\nabla^2 u \tag{2.3}$$

With velocity u, pressure P, kinematic shear viscosity v.

After the implementation of vorticity equation, two main equation which are literally derived from Navier-Stokes equation and continuity equation are as followed (Mat Sahat, M.I , Che Sidek, N.A , 2010)

$$\frac{\delta\Omega}{\delta T} + U\frac{\partial\Omega}{\delta X} + v\frac{\delta\Omega}{\delta Y} = \frac{1}{Re}\left(\frac{\delta^2\Omega}{\delta X^2} + \frac{\delta^2\Omega}{\delta Y^2}\right) \quad (2.4)$$

$$\frac{\delta^2\Psi}{\delta X^2} + \frac{\delta^2\Psi}{\delta Y^2} = -\Omega \quad (2.5)$$

The Ω represent the vorticity and ψ is stream function. Re is the Reynolds numbers of the flow in the cavity and T is the time.

2.2 LATTICE BOLTZMANN

In the last one and a half decade or so Lattice Boltzmann Method (LBM) has emerged as a new and effective approach of computational fluid dynamics (CFD) and it has achieved considerable success in simulating fluid flows and heat transfer (D.A. Parumal, A.K Dass, 2009).

The lattice Boltzmann method allowed particles to move on a discrete lattice and local collisions conserved mass and momentum. Unlike than continuum field approach, kinetic theory assumes that a fluid is made of a huge number of molecular constituents, whose motion obeys Newtonian mechanics. Directly solving the system with a large number of degree of freedom, which is in the order of the Avogadro's number (10^{23}), is impossible. The movement of every individual molecular is not in the current purpose to be discussed, but one molecular is interested in the collective behavior of such system. The statistical description of the system will become predictable. The statistical approach provide a bridge between the macroscopic realm of hydrodynamics and the microscopic realm of atoms and molecular.

Because of the continuity and Navier Stokes equations are only continuous forms of the mass and momentum conservation statements and method that locally conserves mass and momentum will follow some kind of continuity and Navier Stokes equations and it was shown that the lattice gas methods could be used to simulate (rather noisy) hydrodynamics.

However, the lattice gas methods had several drawbacks consisting mainly of their noisy nature and the appearance of some additional terms in the Navier Stokes level equations that limited their success. It was then discovered that instead of discrete particles a density distribution could be adevecting which eliminated the noisiness of the method and allowed for a more general collision operator. This is the lattice Boltzmann method which has been extraordinarily successful for many applications including turbulence, multi-component and multi-phase flows as well as additional applications.

2.2.1 Classical Boltzmann Equation

A statically description of a system can be in term of the distribution function $f(x,c,t)$ where $f(x,c,t)$ is define such as $f(x,c,t)dxdc$ is the number of partical whose position and velocities are essentially dx and dc at time t. If there were no collision, then a short time Δt later each particle would move from x to $x + c\Delta t$ and each particle velocity would change from c to $c + a\Delta t$, where a is the acceleration due to external forces on a particle at x with a velocity c. The number of molecules $f(x,c,t)dxdc$ when there is no collision is equal to the number of molecules $f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc$, therefore

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x,c,t)dxdc = 0 \tag{2.6}$$

However the collisions do occur between the molecules there will be a net difference between the molecules $f(x,c,t)dxdc$ and the number of molecules $f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc$. This can be expressed by

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x, c, t)dxdc = \Omega(f)dxdcdt \quad (2.7)$$

Which is $\Omega(f)dxdcdt$ is the collision operator. On dividing by $dxdcdt$, and letting dt tends to zero gives the Boltzmann equation for f

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a} = \Omega(f) \quad (2.8)$$

2.2.2 Boltzmann Collision Function

Any solution of the Boltzmann equation, $\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a} = \Omega(f)$ required that an expression for the collision operator $\Omega(f)$. If the collision is to conserve mass, momentum and energy it is required that

$$\int \begin{bmatrix} 1 \\ c \\ c^2 \end{bmatrix} \Omega(f)dc = 0 \quad (2.9)$$

Collision can change the distribution function $f(x, c, t)$ in two ways;

i. Some particles originally having velocities c will have some different velocity after collision. This causes in $f(x, c, t)$.
ii. Some particles have other velocities may have the velocity c after a collision, increasing $f(x, c, t)$.

The form of the collision function can be found by assuming that

i. Only binary collisions need to be considered (dilute gas)
ii. The influence of container walls may be neglected
iii. The influence of the external force (if any) on the rate of collision is negligible
iv. Velocities and position of a molecule are uncorrelated (assumption of molecular chaos)

Suppose two particle with initial velocities c and c_1 have velocities c' and c_1' after a collision. Since all particles have same mass, conservation of momentum and energy required that

$$c + c_1 = c' + c_1' \tag{2.9}$$

$$\frac{1}{2}|c|^2 + \frac{1}{2}|c_1|^2 = \frac{1}{2}|c'|^2 \frac{1}{2}|c_1'|^2 \tag{2.10}$$

For an elastic collision, the magnitude of the relative velocity is a coalitional invariant

$$|c - c_1| = |c' - c_1'| \tag{2.11}$$

Under all these assumptions, the Boltzmann equation takes on following form:

$$\Omega(f) = \iint (f f_1 - f' f_1')g\sigma d\Omega dc' \tag{2.12}$$

Where$f = f(x, c, t)$, $f_1 = f(x, c_1, t)$, $f' = f(x, c', t)$, $f_1' f(x, c_1', t)$, g is the particles relative velocities before the Collision and σ is the scattering cross section.

2.2.3 Bhatnagar-Gross-Krook(BGK) Collision Model

The Boltzmann equation without the external force where

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a}\Big| \text{ (Collision)} \tag{2.13}$$

which is,

$$\frac{\delta f}{\delta t} = \Omega(f) \tag{2.14}$$

represent the change in distribution function per unit time due to collision. The particular interest is in the change in distributing function f in time of order τ_f, the average time

between excessive collision. Assuming that at near equilibrium, the system is closed to local Maxwell-Boltzmann state. Moreover, the post-collision distribution function, $f''s$, should be closer to equilibrium than the pre-collision $f's$, because of H- theorem. The distribution function f can be related to the equilibrium distribution function f^{eq} via Taylor's series expansion

$$f^{eq}(x,c,t)f \approx f(x,c,t) + \frac{\delta f}{\delta t}\Big|(collision)\ (\delta t) + o(\delta t)^2 \tag{2.15}$$

$$\frac{\delta f}{\delta t}\Big|(collision) = \frac{f^{eq}(x,c,t)f - f(x,c,t)}{\delta t} = \frac{f^{eq}(x,c,t)f - f(x,c,t)}{\tau_f} \tag{2.16}$$

Where the small time interval δt have ben replaced by the characteristic time between collisions τ_f. This model is frequently called collision model after Bhatnagar, Gross and Krook who first introduced.

2.2.4 The Lattice Boltzmann Equation

The Boltzmann equation with BGK collision model can be expressed as

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} = \frac{f - f^{eq}}{\tau_f} \tag{2.17}$$

That is well known as the BGK Boltzmann equation. The Maxwell-Boltzmann equilibrium distribution function is define as

$$f^{eq} = \rho\left(\frac{1}{2\pi RT}\right)^{d/2} exp\left\{-\frac{(c-u)^2}{2RT}\right\} \tag{2.18}$$

The BGK lattice Boltzmann equation can be derived by further discretise using an Euler time step in time step in conjunction with an upwind spatial discretization and then setting the grid spacing divided by the time step equal to the velocity

$$\frac{f(x,t+\Delta t)-f(x,t)}{\Delta t}+c\frac{f(x+\Delta x,t+\Delta t)-f(x,t+\Delta t)}{\Delta x}=\frac{f-f^{eq}}{\tau_f} \tag{2.19}$$

$$\frac{f(x,t+\Delta t)-f(x,t)}{\Delta t}+c\frac{f(x+\Delta x,t+\Delta t)-f(x,t+\Delta t)}{\mathrm{c}\Delta x}=\frac{f-f^{eq}}{\tau_f} \tag{2.20}$$

As a result

$$f(x+c\Delta t,t+\Delta t)-f(x,t)=-\Delta \mathrm{t}(\frac{f-f^{eq}}{\tau_f}) \tag{2.21}$$

The LBGK model with single relaxation time, which is a commonly used lattice Boltzmann method, is given by (D.A. Parumal, A.K Dass, 2009)

$$f_i(x+c_i\Delta t,t+\Delta t)-f_i(x,t)=-\frac{1}{\tau}[f_i(x,t)-f_1^0(x,t)] \tag{2.22}$$

Where f_i is the particle distribution function, $f_1^0(x,t)$ is the equilibrium distribution function at x, t, c_i is the particle velocity along the ith direction and τ is the time relaxation parameter. The D2Q9 square lattice used here has nine discrete velocities. A square lattice is used, each node of which has eight neighbors connected by eight links as shown in Fig. Particles residing on a node move to their nearest neighbors along these links in unit time step. The particle velocities are defined as

$$C_i=o \qquad i=0 \tag{2.23}$$

$$C_i=\left(\cos\left(\frac{\pi}{4(i-1)},\sin\left(\frac{\pi}{4(i-1)}\right)\right)\right),i=1,2,3,4 \tag{2.24}$$

$$C_i=(\cos\left(\frac{\pi}{4(i-1)}\right)\sin(\frac{\pi}{4(i-1)}))\,,i=5,6,7,8 \tag{2.25}$$

The macroscopic quantities such as density ρ and momentum density ρu are obtained as velocity moments of the distribution function f_i as follows:

$$\rho = \sum_{i=0}^{N} f_i \tag{2.26}$$

$$\rho u = \sum_{i=0}^{N} f_i C_i \tag{2.27}$$

Where N = 8. In the D2Q9 square lattice, a suitable equilibrium distribution function that has been proposed is

$$f_i^{(0)} = w_i \rho \left[1 - \frac{3}{2}u^2\right], \text{ i=0} \tag{2.28}$$

$$f_i^{(0)} = w_i \rho [1 + 3(c_i u) + 4.5(c_i u)^2 - 1.5u^2], \text{ i=1, 2, 3, 4} \tag{2.29}$$

$$f_i^{(0)} = w_i \rho [1 + 3(c_i u) + 4.5(c_i u)^2 - 1.5u^2], \text{ i=5, 6, 7, 8} \tag{2.30}$$

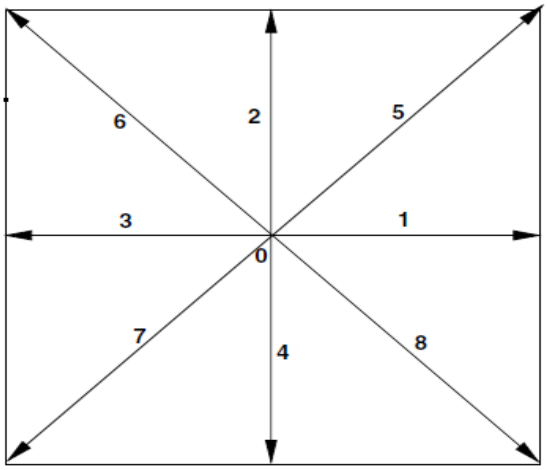

Figure 2: D2Q9 lattice and velocities

Source: D.A. Parumal, A.K Dass, 2009

Where the lattice weights are given by w_0 = 4/9, w_{1-4} = 1/9 and w_{5-8}= 1/36. The relaxation time which fixes the rate of approach to equilibrium is related to the viscosity by (S.Hou et al, 1995)

$$\tau = \frac{6v+1}{2} \tag{2.31}$$

The units for v are lu^2ts^{-1}. Note that τ >1/2 for positive (psychical) velocity. Numerical difficulties can arise as τ approaches ½. A value of τ = 1 is safest and lead to $v = \frac{1}{6}lu^2ts^{-1}$.(M.C. Sukop et al, 2005)

2.3 DISCRETISATION OF LBM

Discretisation is the process of dividing into a finite number of elements a continuum object. The lattice Boltzmann method start from the following Boltzmann equation for discrete velocity distribution in two and three dimensions

$$\frac{\delta f_i}{\delta t} + c_i.\nabla f_i = \Omega(f_i) \tag{2.32}$$

A commonly used LBM is the so-called lattice BGK model where the collision $\Omega(f_i)$ is replaced by the BGK collision model

$$\frac{\partial f_i}{\partial t} + c_i.\nabla f_i = -\frac{1}{\tau_f}(f_i - f_i^{eq}) \tag{2.33}$$

Where the collision is assumed to lead the particle distribution function relaxes to its equilibrium state at a constant rate. If the time derivative is replace by a first order time difference, first order upwind space discretization is used for the convective term $c_i.\nabla f_i$ and $\Delta x = \Delta t = 1$ is set, the discretised version of lattice Boltzmann equation is obtained

$$f_i(x + c_i, t_1) - f_i(x, t) = -\frac{1}{\tau_f}(f_i - f_i^{eq}) \tag{2.34}$$